International Journal of Automobile Engineering Research and Development (IJAuERD)
ISSN (P): 2277-4785; ISSN (E): 2278-9413
Vol. 9, Issue 1, Jun 2019, 1-10

TRANS STELLAR
•Journal Publications •Research Consultancy

THE SIMULATION OF THREE-DIMENSIONAL HEAT TRANSFER THROUGH AN INSULATED PIPE BY VARIATION OF SOLVER PARAMETERS

NISHANT SHARMA. R[1], AAYUSH CHAWLA[2] & HITHESWAR. R[3]

[1,2]Student, Vel Tech University, Chennai, Tamil Nadu, India

[3]Student, Velammal Institute Of Technology, Chennai, Tamil Nadu, India

ABSTRACT

Conjugate heat transfer (CHT) involves three types, namely conduction, convection and radiation. Where conduction is the fastest mode of heat transfer. This formulation of heat transfer is governed by set of heat equations consisting of conformity with a set of patterns of two different systems namely body domain (solid) or Fluid Domain. The body domain is governed by set of steady state two dimensional or three-dimensional conduction for thin bodies which is given by Laplace equations. In a fluid domain under laminar flow we use Navier-Strokes Energy Equation for Boundary layers which are large, Reynolds Number and Prandtl Number are the two major factors influencing the flow. In our model we have used turbulent flow of the fluid inside the interface which is simulated by Reynolds Average Navier-Stroke (RANS) energy equation, Boundary layer equations are large values of the Reynolds number. Here we use initial boundary and conjugate conditions specifying the spatial distribution of variable dynamic and thermal equation with time respectively in no slip conditions. Here we have used Converge CFD Cygwin Para view for the complete conjugate heat transfer of the insulated pipe.

KEYWORDS: CHT, Heat Transfer, Navier Stroke Equation, RANS, Boundary Layer, Laplace, Reynolds Number, Prandtl Number, Spatial Distribution, No Slip & Converge CFD Cygwin Paraview

Received: Dec 13, 2018; **Accepted:** Jan 03, 2019; **Published:** Jan 23, 2019; **Paper Id.:** IJAuERDJUN20191

Original Article

1. INTRODUCTION

A three-dimensional model is developed to investigate the heat transfer through a solid coating over a steel pipe. The design is constructed in Converge CFD. The design of the model is constructed in Converge CFD, Cygwin and Para view. Converge CFD, is a revolutionary software that eliminates the grid generation bottleneck from the simulation process. Cygwin is a Linux interface simulation software and Para view is an open source, multi-platform data analysis and visualization application. Then the model is simulated using various case setups and their conduction and convection data's are obtained and corresponding results of heat transfer rates are obtained through graphs and discussed

2. LITERATURE SURVEY

- Three-dimensional conjugate heat transfer in the micro channel heat sinks for electronic packaging by Andrei G. Fedorov, Raymond Viskanta

  A three-dimensional model is developed to investigate flow and conjugate heat transfer in the micro

channel-based heat sink for electronic packaging applications. The theoretical analysis performed, provides a fundamental understanding of the combined flow and conjugate convection – conduction heat transfer in the three-dimensional micro channel heat sink. The model formulation is general and only a few simplifying assumptions are made.

- Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries by Dieter Bohn, Jing Ren, Karsten Kusterer

The formation of kidney vortices can significantly be reduced by shaped holes instead of cylindrical holes. Numerical methods are applied over a duct flow with cooling fluid injection through different hole configurations. Conjugate heat transfer and adiabatic heat transfer calculation are applied on the wall half way on both the sides. It is understood from the experiment that for shaped configuration the secondary flows are significantly low than the cylindrical holes. From various shaped configurations we come to a conclusion that fan shaped configuration is up to three times effective than the cylindrical hole configuration.

3. DESIGN STAGE

A cylindrical pipe is constructed in Converge CFD using the construct tools. It is placed to the negative of the base plane. The geometric bounding boxes are set as X: -2.00E-002 to 2.00E-002; Y: -1.991468E-002 to 1.991468E-002; Z: 0.00E+000 to 2.00E-001. Length in X (m): 4.000000E-002; Length in Y (m): 3.982937E-002; Length in Z (m): 2.000000E-001. The cylindrical is constructed with insulation and the insulation material chose is aluminium. With thickness 0.00852 (m).

4. PROCESS STAGE

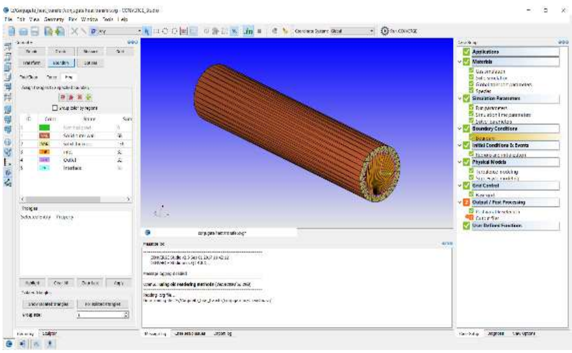

Figure 1

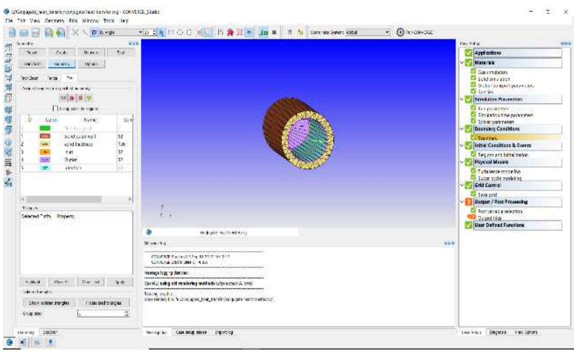

Figure 2

The process starts with setting up various case setups in the Converge CFD such as materials, simulation parameters, Boundary conditions, initial conditions & events, physical models, grid controls, output/post processing. Using these case setups, we will be able to define various test values regarding the simulation process and generate the results of the simulation.

4.1 Various Case Setups

(i) Materials

- **Gas Simulation**

In this case setup we choose the equation of gas state as Redlich Kwong and the critical temperature and pressure as 133.0 K and 3.77e+06 Pa.

- **Solid Simulation**

The solid coating selected here is aluminium whose melting point is 1700.00K for which 171 rows of different Temperature[k] values are to be inserted in the value table with their Density [kg/m^3], Specific heat [J/kg*k], Conductivity[w/(m*k)]. The initial and final value for the data is given as 0.000000+00K; 2.70000e+03 kg/m^3; 9.03000e+02 J/(kg*k); 2.37000e+02 W/(m*k) and 1.70000e+03 K; 2.70000e+03 kg/m^3; 9.03000e+02 J/(kg*k); 2.37000e+02 W/(m*k).

- **Global Transport Parameters**

In this case setup we will be setting up the parameters used in the simulation process. The Turbulent Prandtl number is set as 0.9 and Turbulent Schmidt number is set as 0.78.

- **Species**

The species that is selected in this simulation is gas and solid species. The gases such as O2 and N2 are used as inlet gas flowing through the solid pipe and the solid species is selected as Aluminium.

(ii) Simulation Parameters

- **Run Parameters**

In this run parameters we tend to set up the solver as Transient, simulation mode as Fully Hydrodynamic and Gas flow solver as Compressible.

- **Simulation Time Parameters**

Setting up the parameters available in this process helps us to direct the flow of air into the solid pipe in various ways of motion. Minimum and Maximum time-step is set as 1e-07s and 1.0s. Variable time-step algorithm is selected in time-step selection. The maximum convection CFL limit is 1.0 and maximum diffusion CFL limit is 2.0. Droplet motion time-setup control multiple is set as 1.5 and droplet evaporation time-step multiple as 9999.0

- **Solver Parameters**

In solver parameter we set the PISO parameters section. The input values given for PISO convergence criterion multiplier, minimum and maximum number of PISO iterations and PISO tolerance are 20.0, 2, 9, 1e-03.

(iii) Boundary Conditions

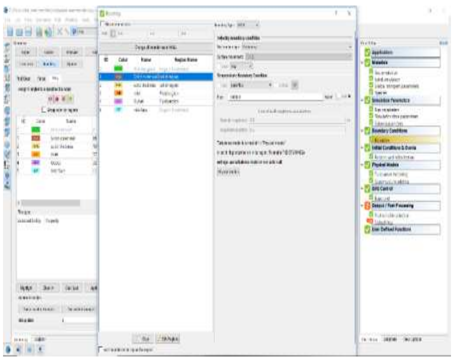

Figure 3

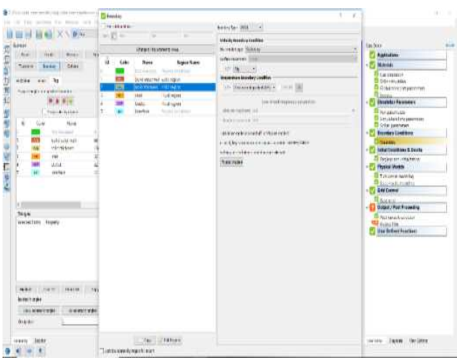

Figure 4

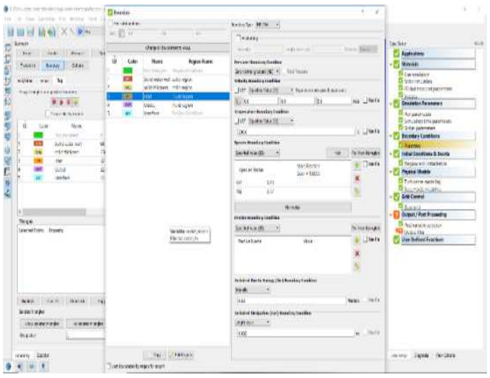

Figure 5

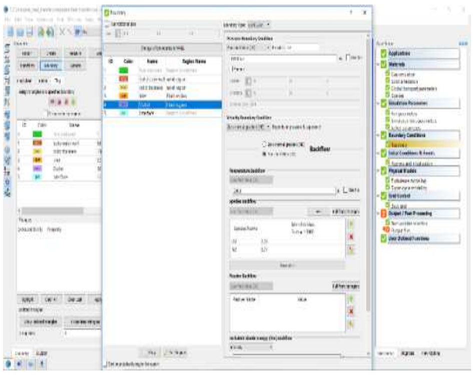

Figure 6

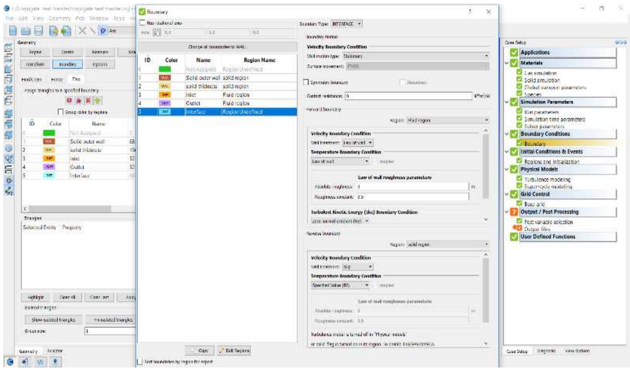

Figure 7

(iv) Initial Conditions & Events

- **Regions and Initialization**

There are two types of regions, namely fluid region and solid region.

- **Fluid Region**

For fluid region the Stream ID is set as 0. The temperature and pressure are set as 300.0 K and 101325.0 Pa. The species selected is air and their composition are O2 and N2. Their respective Mass Fraction is 0.23 and 0.77.

- **Solid Region**

For fluid region the Stream ID is set as 1. The temperature and pressure are set as 300.0 K and 101325.0 Pa. The species selected is aluminium and its mass fraction is 1.

(v) Physical Models

- **Turbulence Modelling**

In Turbulence modelling Reynolds Averaged Navier-Strokes (RANS) is selected for this type of simulation. Various constant values such as VON Karman's constant: 0.42 Law of the wall parameter: 5.5 in Heat Model. And corresponding constant values are given for RANS constant also.

- **Super-Cycle Modelling**

In this section data's are given for all the required fields. Begin storing supercycle data is given as 0.05. Solid side heat transfer solver is set as steady solver. Time length for each cycle stage is given as 0.05. Total number of stages is

given as 1. Conduction CFL number is provided as 100.0. Sensible Internal Energy (SIE) Tolerance is set as 1e-0.7 and relaxation factor is given as 1.4. The output points x, y, z is given as -0.012933, 0.01179, 0.18.

(vi) Grid Control

In base grid control the base grid size for dx, dy, dz are given as 0.004 m.

(vii) Output/ Post Processing

- **Post Variable Selection**

In this section typical parameters such as Density, Pressure, Temperature, Velocity are selected and Region ID is selected in Geometry/Location.

- **Output Files**

Turn on the Generate mixing related output option and set the respective time interval for various data file options.

5. RESULT AND DISCUSSIONS

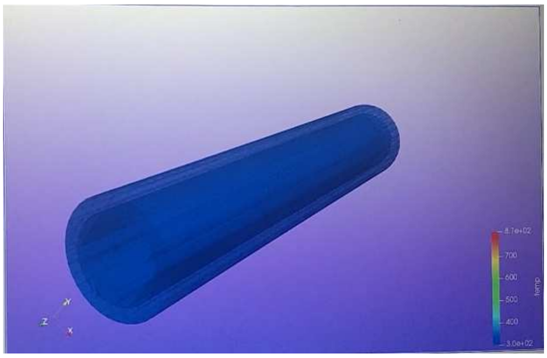

Figure 8

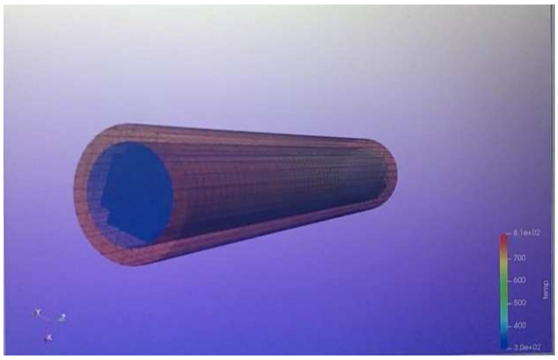

Figure 9

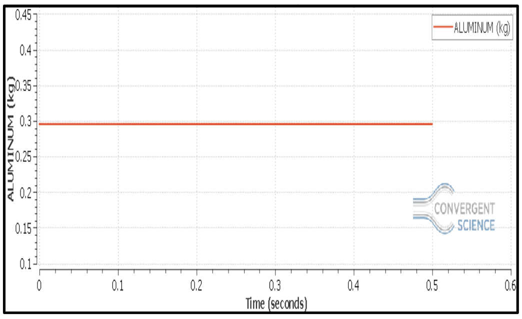

Figure 10

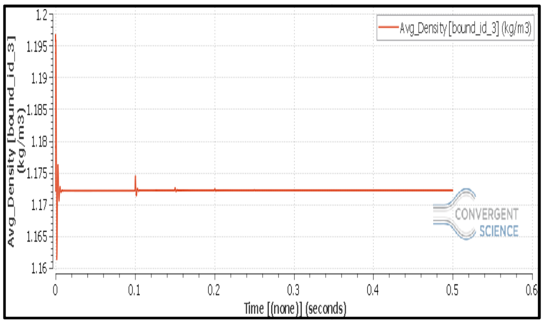

Figure 11

From this graph we can see that the mass of the aluminium is constant with respective to time. There is no deflection in the mass of the insulation.

Average Density[bound_id_3] has greater variation at the start of the experiment and later its constant with respect to change in time.

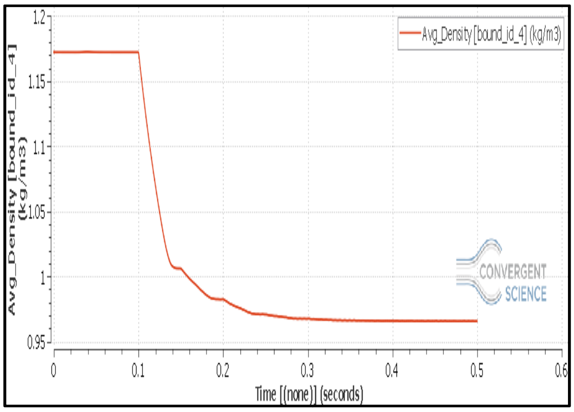

Figure 12

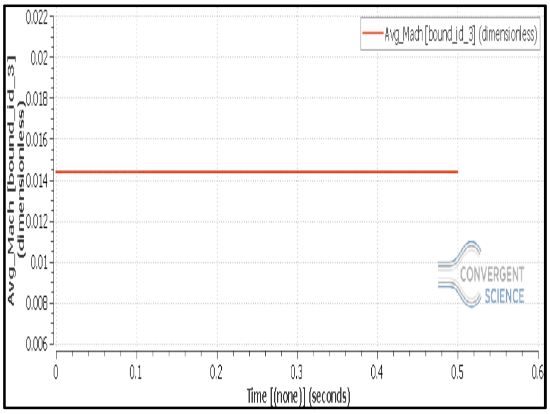

Figure 13

The average density [bound_id_4] is remains constant for a period of time and shows a sudden decline in its density rate and remains constant over a period of time.

From the above graph we may come to a conclusion that the Avg. Mach[bound_id_3] is constant with respect to change in time interval.

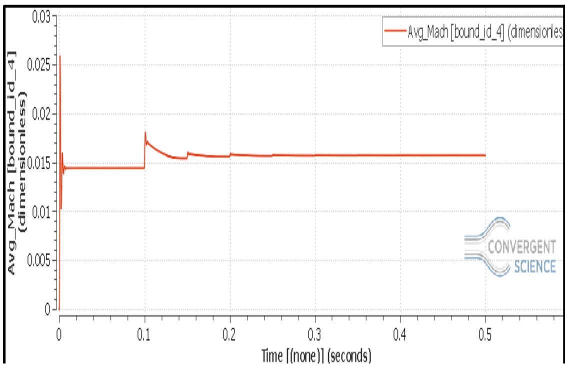

Figure 14

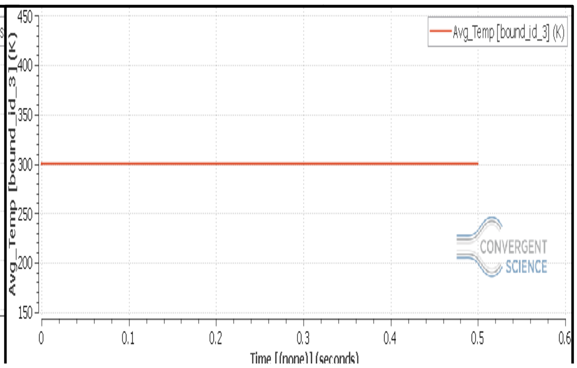

Figure 15

In Avg. Mach[bound_id_4] vs Time interval we can see to it that there is huge increase in the Avg. Mach rate within a few milliseconds of the start and becomes constant only after a few seconds later.

The Avg. temp [bound_id_3] remains constant with respect to time throughout the experiment from the beginning

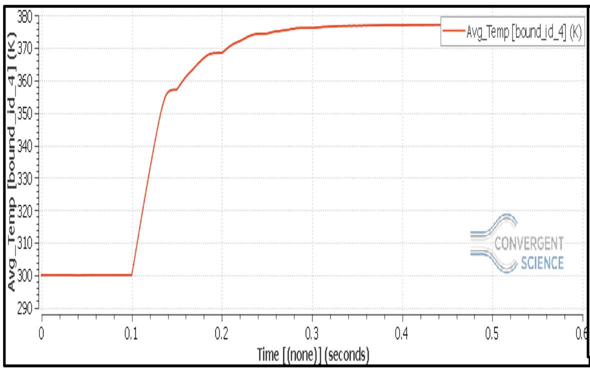

Figure 16

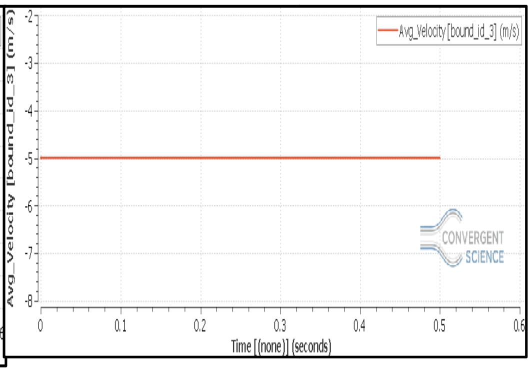

Figure 17

The Avg. temp [bound_id_4] remains constant only for a few seconds in the beginning and later on shows a steep increase at a point and further increases gradually and reaches the steady state.

The Avg. velocity [bound_id_3] remains constant with respect to time from the start of the heat transfer through the pipe.

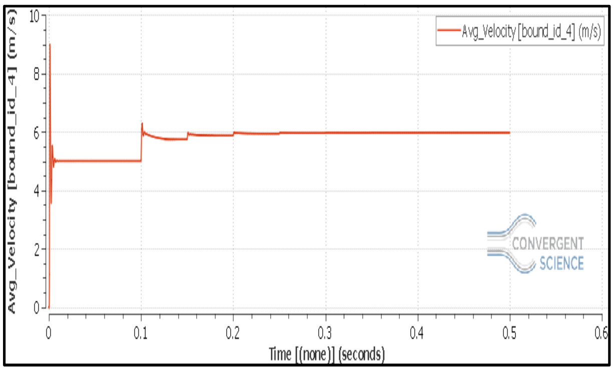

Figure 18

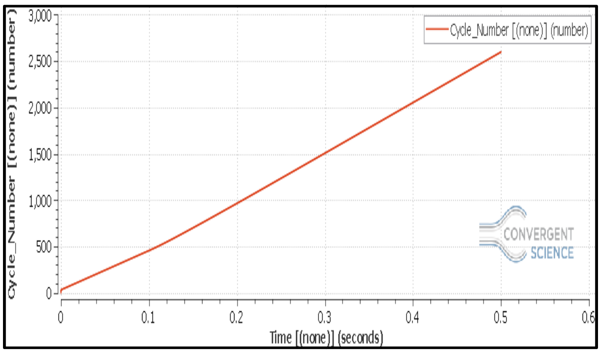

Figure 19

The Average velocity [bound_id_4] show a higher increase in velocity at the start and later on maintaining a velocity constant after 0.2 seconds.

The cycle number shows a steady increase in rates from the start with respect to the time interval.

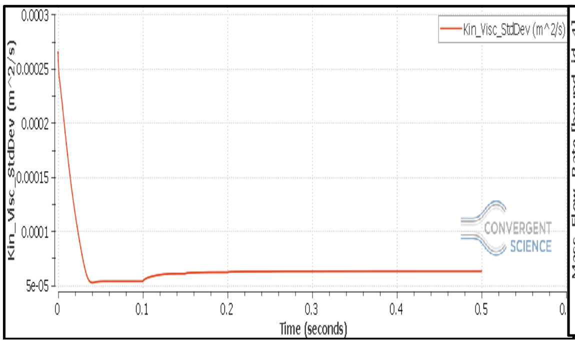

Figure 20

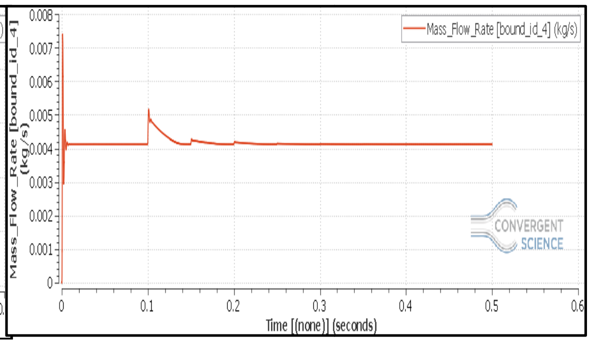

Figure 21

The Kin_Visc_Std Dev shows a steady decline for a few seconds and later shows some deviation with a period of time and attends a constant deviation rate after a few seconds of the start.

The Mass_flow_rate [bound_id_4] shows a high deflection at the beginning and later a small deflection between 0.004(kg/s) and 0.005(kg/s) and remains constant at 0.0041(kg/s) after a period of time.

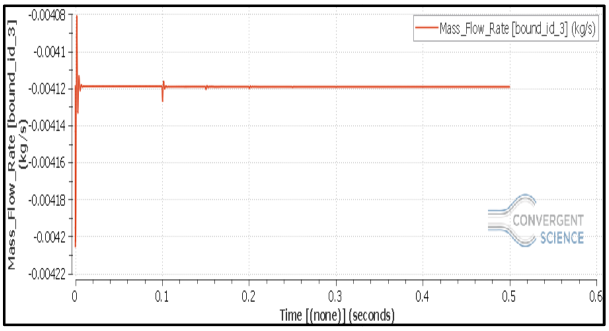

Figure 22

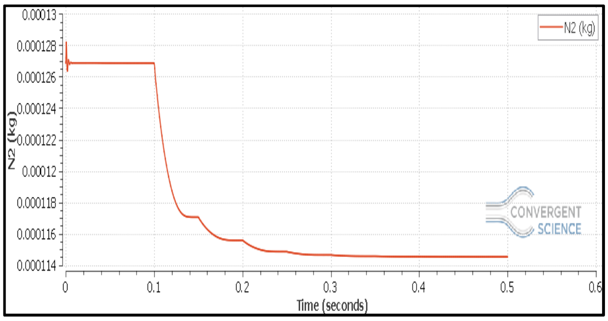

Figure 23

The Mass_flow_rate [bound_id_3] shows a high deflection at the beginning and later remains constant at -0.00412(kg/s) after a period of time.

The gas N2 shows a decline in its mass from 0.000128Kg to 0.000114kg after a period of time. After a certain period of time it remains constant and again decreases and remains constant after a few seconds.

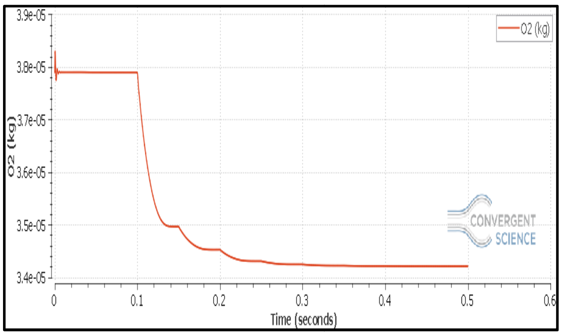

Figure 24

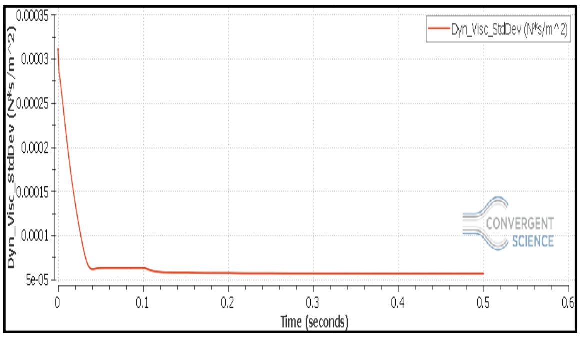

Figure 25

The O2 gas shows the same behaviour as N2 gas. It also shows in a gradual decrease of its mass after a few seconds of start and remains constant.

The Dyn_Visc_std Dev shows a decrease in its behaviour and remains constant after few seconds when its reduced to a point of 5e-05(Ns/m^2).

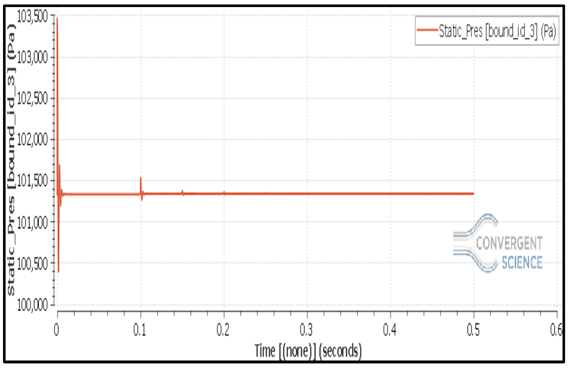

Figure 26

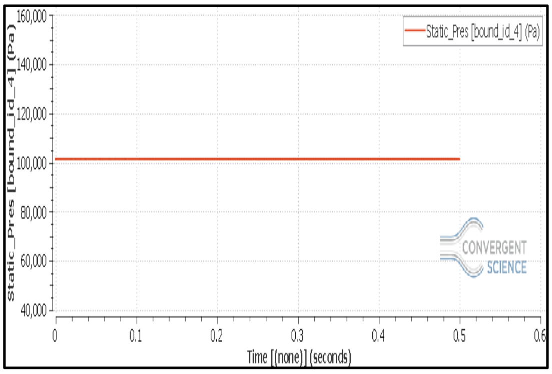

Figure 27

The static pres [bound_id_3] shows a high increase in the start and later shows only slight deflections and remains constant at 101,400 Pa.

The static pres [bound_id_4] shows no change in its pressure and remains constant with respect to time from the start.

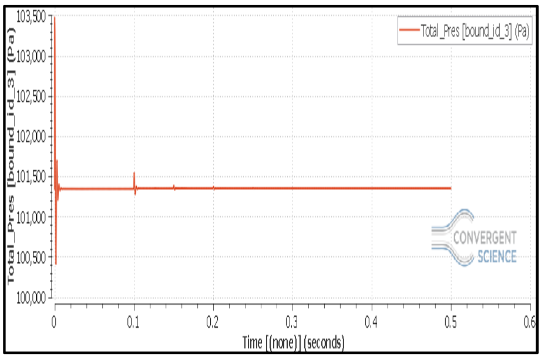

Figure 28

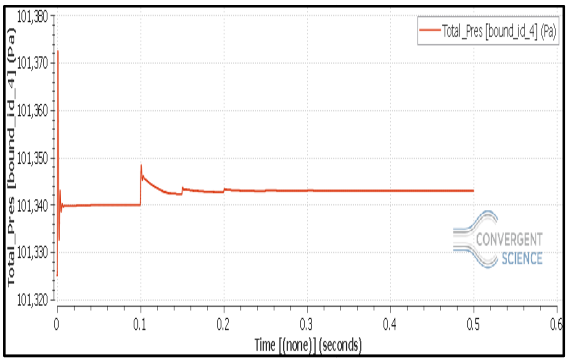

Figure 29

The Total_pres [bound_id_3] shows a high increase in the start and later shows only slight deflections and remains constant at 101,400 Pa.

The Total_pres [bound_id_4] show a great increase in its pressure at the start and remains constant for a period of time and again shows a slight deflection and remains constant throughout the period of the time till the end.

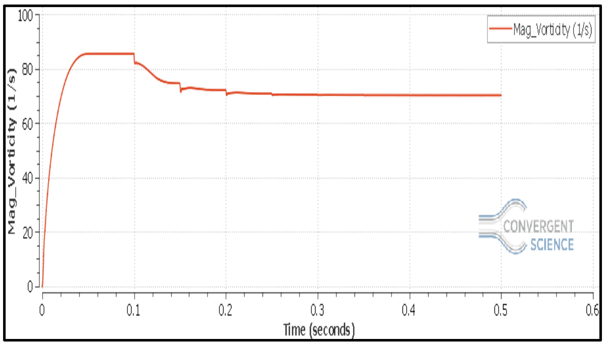

Figure 30

The Mag_Vorticity increases gradually over a few seconds at the beginning and shows a decline over a period of time and finally comes to a constant state where there is no deflection of the mag_vorticity after a period of time.

6. CONCLUSIONS

From the above figures we get to a conclusion that the heat distribution is in the order of convection, conduction and radiation. The method of simulating the body fluid conjugate heat transfer simulation is approximately equal to the method of conjugation body fluid solution i.e. numerical methods. The analytical reduction to the conduction problem is well known as Duhamel's integral for the heat flux inside a solid pipe with arbitrary variable temperatures is a sum of series of temperature derivatives for various boundary conditions that can be evaluated by the method of standard deviations across the inlet and outlet boundaries. The variation of velocity and temperature is represented in fig (8 & 9). Condition of the boundary domains for the specifying temperatures distribute the heat flux in the positive Z direction where the heat is rejected from the interface. The heat flux which is distributed from the interface to the outer solid wall is represented in the figure. Thus, we have dealt with conjugate heat transfer through a solid steal pipe insulated by a coating of aluminium over it and concluded the various heat transfer results and represented the graphical parameters for the physical parameters.

REFERENCES

1. *Andrei G. Fedorov, Raymond Viskanta, Three-dimensional conjugate heat transfer in the micro channel heat sink for electronic packaging, in: International Journal of Heat and Mass Transfer 43 (2000) 399–415.*
2. *Kumar, R., Pedgopu, V., Kumar, A. N. I. L., Thakur, R. O. B. I. N., & Pundir, A. N. I. L. (2013). CFD based analysis heat transfer and friction characteristics of broken multiple rib roughened solar air heater duct. Int J Mech Prod Eng Res Dev, 3, 165-172.*
3. *Dieter Bohn, Jing Ren, Karsten Kusterer, Conjugate Heat Transfer Analysis for Film Cooling Configurations with Different Hole Geometries, in: Proceedings of ASME Turbo Expo 2003 Power for Land, Sea, and Air June 16 – 19, 2003, Atlanta, Georgia, USA.*

4. *Bodade, P. R., Jogi, N., Gorde, M., Paropte, R., & Waghchore, R. Use of Internal Threads of Different Pitches To Enhance Heat Transfer In A Circular Channel.*

5. *Chaudhari, S. C., Yadav, C. O., & Damor, A. B. (2013). A comparative study of mix flow pump impeller cfd analysis and experimental data of submersible pump. International Journal of Research in Engineering & Technology (IJRET) ISSN, 2321-8843.*